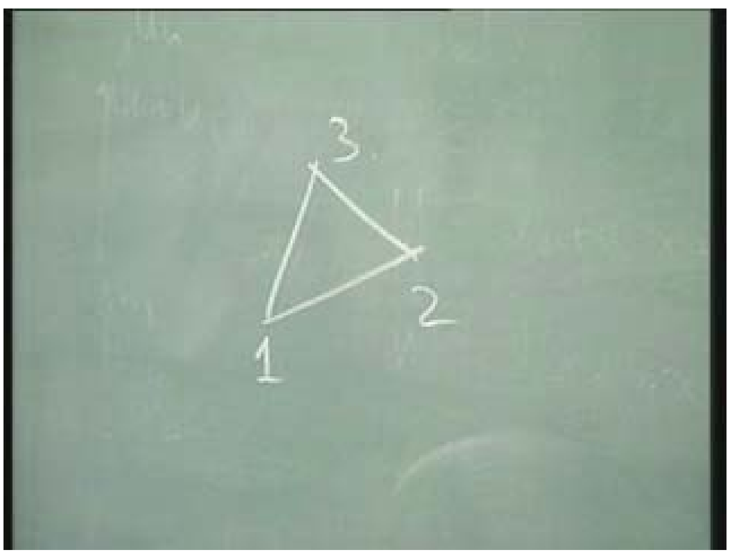
Introduction to Finite Element Method
Dr. R. Krishnakumar
Department of Mechanical Engineering
Indian Institute of Technology, Madras

Lecture - 17

We made some very important conclusions in the last class about how the displacement results behave in a finite element situation and how strains behave and so on. What we are going to do in this class is to quickly look at one or two elements and then look at some practical problems so that you can link up what all the steps we did in the class to what are actually done in a practical finite element software. So, you will be able to understand what we actually do there is based upon the theory which we learnt during these few classes, may be about 10 to 15 classes. We will continue with the theory later in the next class, but let us look at certain examples, simple examples so that we can finish it in the next 50 minutes and see how these results are and how they are related to what we learn during this course. Let us now look at a simple element.

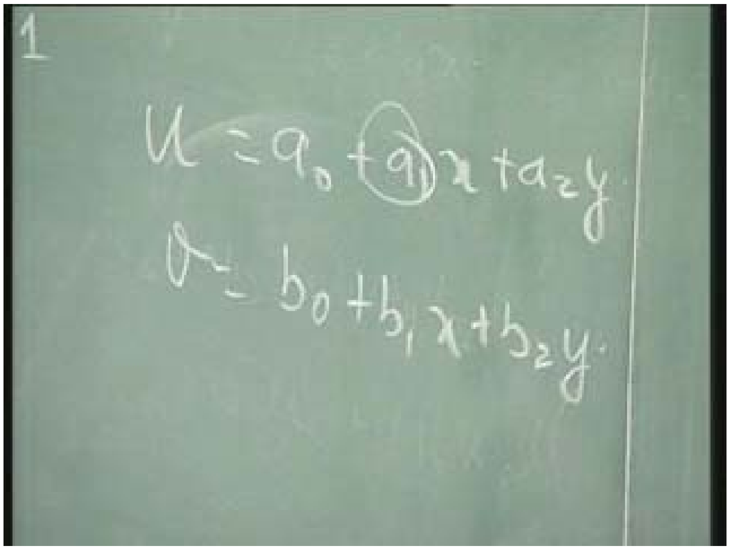
(Refer Slide Time: 2:05)



Say for example let us say that we are going to do a two dimensional problem. Let us look at a simple element, say a triangular element. Triangular element and its counterpart in the three dimensions, it is tetra elements, are extensively used in many of the packages; the triangular elements are extensively used. What is the reason? The reason is that many of the automatic mesh generators, as we call it, automatic mesh generators are written to generate triangular tetra elements. In a two dimensional situation, today we have softwares. Why? Almost all the softwares will be able to create a quadrilateral mesh. But there are not many packages which can generate at the corresponding brick or a hexa element in three dimensions, because mathematically it is a tough problem. It is not yet a solved problem and so, unless you take lot of efforts to generate a mesh which is a brick in the three dimensions. If you want to resort to automatic mesh generation, then it would not be possible to generate a hexahedron element. So, we are stuck many times, especially in 3D, with this kind of three noded and four noded elements in 3D.

Why do I use the word stuck, are there problems with it? Yes, the problems are very obvious. What are the problems? In order to understand what this problem is let us look at the interpolation values. Say for example, let me say that u is the displacement in the x direction then, I can write it down. How do I write it down here?

(Refer Slide Time: 4:07)

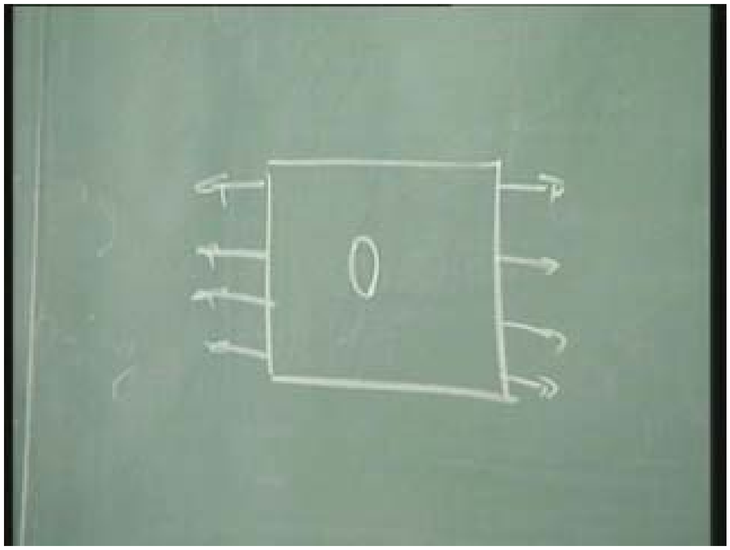


There are three nodes. So, u is equal to say a_0 plus a_1x plus a_2y, the x and y directions. The displacement in the y direction, please note that I am calling displacement in the y direction as v, so v is now written as b_0 plus b_1x plus b_2y. Can someone point out what would be the difficulty if I have something like this? Is it possible? Look at strains now. Epsilon_{11}; what is epsilon_{11}? Dow u by dow x. What would happen to that? In this case it becomes a_1, it becomes a_1 and what happens to dow v by dow y? b_2 and so on. In other words what happens physically? It so happens that the strain is a constant; correct. So the strain becomes a constant in an element. So, these elements are called as constant strain triangle elements. We will see more about these elements later in the course, may be from tomorrow onwards we will concentrate more on these kinds of elements. But, right now it is important to understand that it is possible to understand how the elements would behave? How the elements would behave given a practical situation by understanding how shape functions are written for these elements?

Straight away it comes out from the interpolation from which I can get shape functions and so on. I can write it in terms of u is equal to n_1 u_1 plus n_2 u_2 plus n_3 u_3 and exchange it with a_0 a_1 a_2 and so on. We will not do it right now, but it is very easy. It is not very difficult; we have very special methods to do that. We will resort to those special methods in the next class, in the coming classes, but right now, it is important to understand that this element which is extensively used has the behavior which shows that it has a constant strain throughout the element. What does it mean? It means that if you use this element at a place where there is going to be a variation in stress and strain, then you have to use lot of elements. Then the results would depend totally on the number of elements. What we are going to do in this class is to investigate certain results.

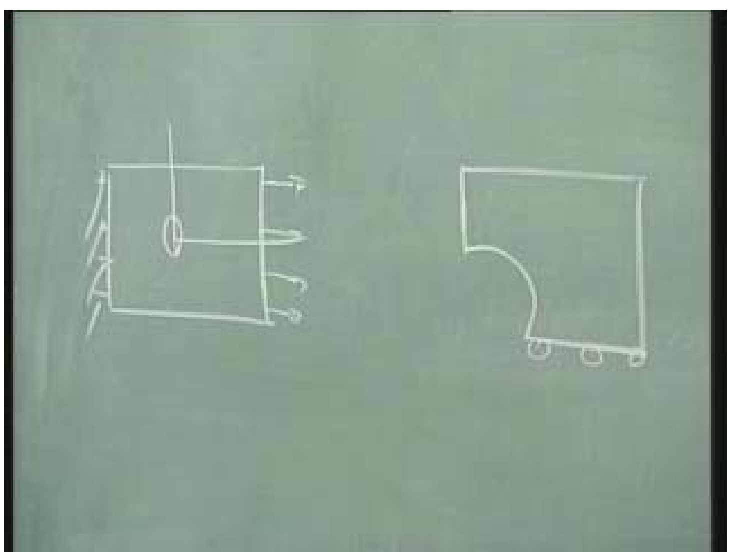
Try to understand what the stress jump is, how it is taken care of in the finite element packages. In order to do that let us look at two problems. What are the two problems?

(Refer Slide Time: 7:20)



Let us say that I take a sheet like this, put a hole in the centre and I want to pull this. As you know, I have to give a boundary condition; I cannot just pull it like this. So, I do it by two means. One is by fixing this side, the left hand side I am going to fix.

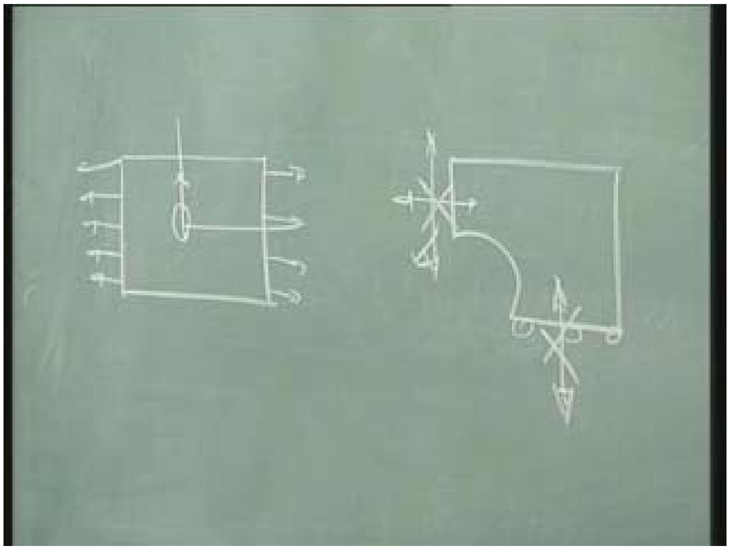
(Refer Slide Time: 7:48)



I will fix this, pull it. Let us assume that the material is steel and then we will do this problem. Let us now look at displacements and so on. The problem here is that you may immediately argue that this problem is not the same as what I had in mind, because I wanted to pull on either side. This we have already seen, but I have fixed it at one side. We will do another variation of this problem by taking symmetry about these planes, about the center planes. I will have a problem which will be like this, where it is very clear that these have no displacements or the material particles which are sitting at the symmetry line have no displacement perpendicular to the plane of symmetry. The name itself shows that it is symmetrical, which means that these points, material particles which are sitting in those planes, which depict symmetry have no displacement. Please note that in order to have symmetry we should have not only loads to be symmetrical, not only geometry to be symmetrical, but also material to be symmetrical.

In other words, if I take a particular plane, I should be able to reflect both-? the geometry, load as well as the material about it and get the other half. So, there can be more than one plane of symmetry. In this particular problem for example, we have two planes of symmetry. We have one plane here and one plane there. I reflect it about this plane and I reflect it about this plane to get the complete picture. Please note, by symmetry what I am doing is not this problem where I have fixed it one side. It is not that problem I am doing. I am doing my original problem, where I have loads applied on either side. So, that is what the problem I am doing.

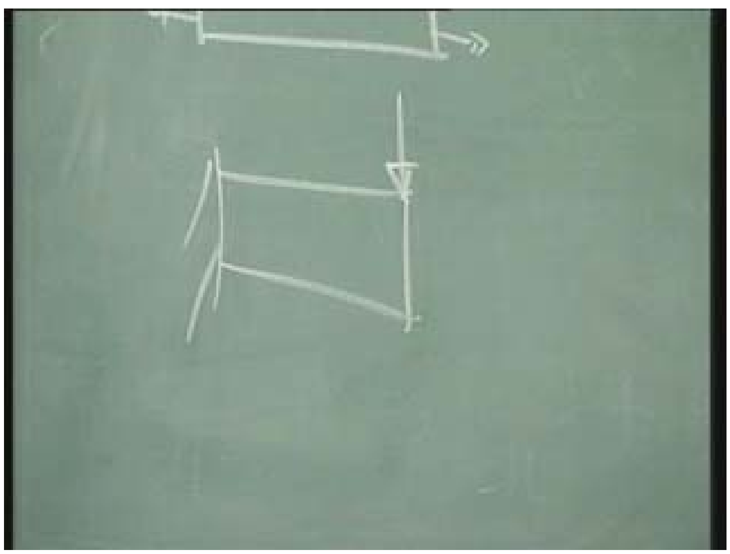
(Refer Slide Time: 10:11)



If you look at actually a material particle, which is sitting at that place and go back to my old problem where I apply loads here like this, if I look at a material particle that is sitting there it is symmetrically loaded on either side, so, he has no chance to move from the plane and hence what do we do? We fix it or we fix the displacement of those material particles which are sitting on the symmetric plane in such a fashion that it will not get displaced perpendicular to the symmetric plane. So, this point will not move. So these displacements, I would fix. In the same fashion, I will also fix displacements perpendicular to this symmetric plane. On the other hand, symmetry plane does allow displacement in this direction; that direction is allowed, it is free to move in that direction, unless I apply load in the bi axial sense. Since this is only uni axial sense, I can apply load on either side. Then, yes, there would be symmetry even at that position.

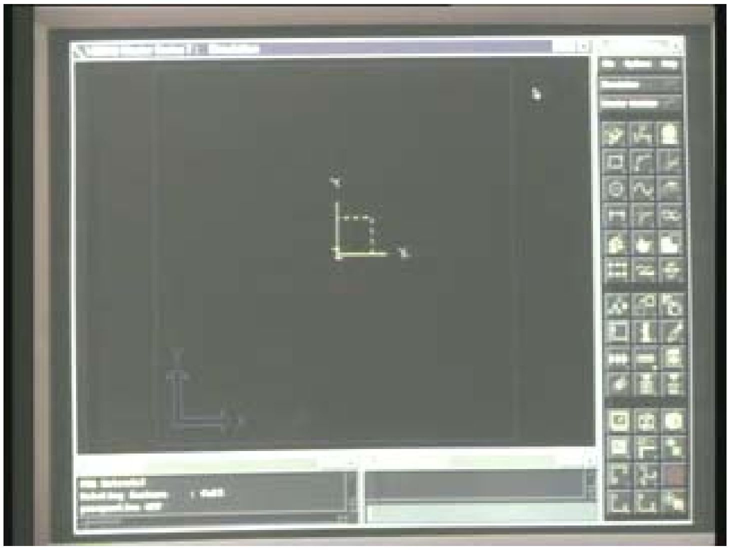
But right now, it is not there and hence it is very nicely depicted with no displacement perpendicular to the boundary condition. We will talk about symmetry again, but let us look at certain practical problems so that you can understand and put things in proper prospective as to what all we have done.

(Refer Slide Time: 12:02)



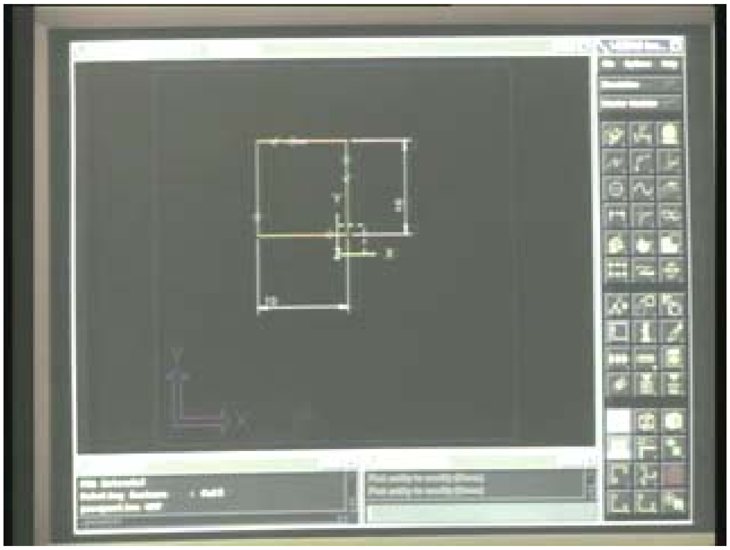
What we are going to do is to work out these problems and may be one more problem we will take up, a beam problem, a simple beam problem and then apply a load here. Then triangulate this beam with different densities of triangle; may be first we will have about 10 or 15 triangles and then may be more and so on. Let us again look at the results. We will start comparing, keep your notes ready so that let us look at the values and may be write down the maximum values and compare. So, let us get back to the computer and see how we are going to do these problems. You are all ready with your pens? Let us now get to the computer.

(Refer Slide Time: 12:52)



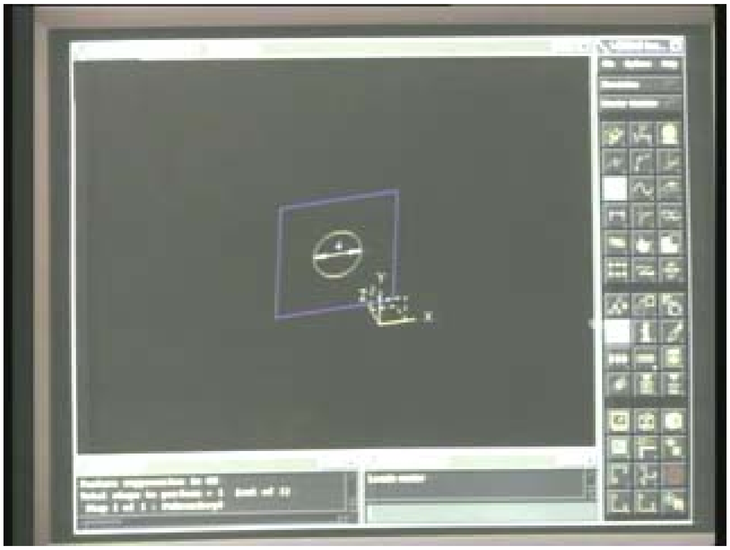
The package which we are going to use is called as ideas. This is from a company called S D R C. It is one of the popular CAD CAM packages. There are other packages like pro engineer and catia, unigraphics and so on. But, this particular package along with certain other packages like pro-e has an integrated CAD CAM software, a part of which is finite element as well. There are other very specialized softwares that are available for finite element only, for example Anses, MSC Nastran, Abacus, Nisa. These are other packages which are for doing finite element analysis. What we are going to do is to use this package to demonstrate as to how exactly we are going to do a finite element problem. So, the first step in our finite element problem is to draw the region of interest.

(Refer Slide Time: 14:04)



In this case it is going to be just what we saw that square with the circle at the center. Let us see how we can draw that particular problem? What he is trying to do, Mr. Andrew, who is operating this, what he is trying to do is to first draw a poly line. Let us see what he does. There is what is called as a dynamic navigator. As he does it, it shows perpendicularity and all that other things are also shown. He has now drawn the poly line and he is dimensioning it for our sake and giving a name to this as well. He is just changing it so that it becomes a square of 10 by 10.

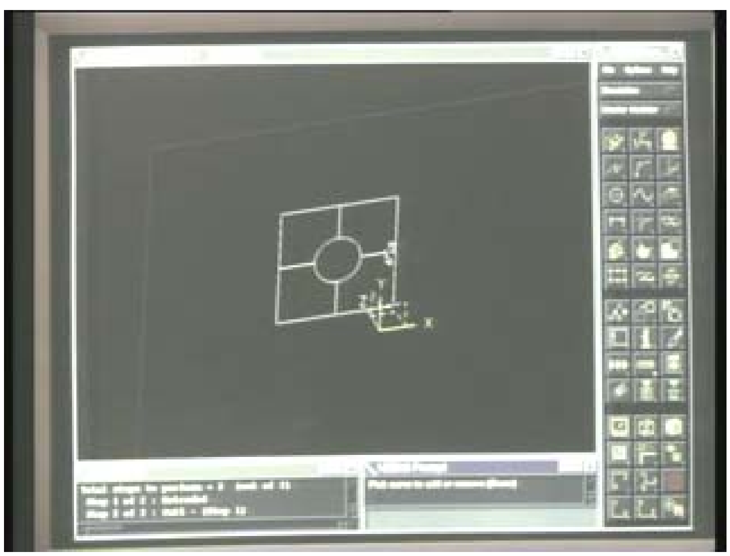
(Refer Slide Time: 15:00)



The next step he is going to do is to draw a circle at the center. Before we go to that circle, let us make this as a surface, because it is just a line. Then we have to make this a surface, because we are going to apply plane stress condition. We will make it as a surface and so he has now made it. He has joined the poly lines or he says that whatever is inside this poly line is a surface. That is what he did in the second step. Let us see that he puts a circle. How does he put a circle? May be he will extrude out or cut out a small circle at the center let us see how he does it.

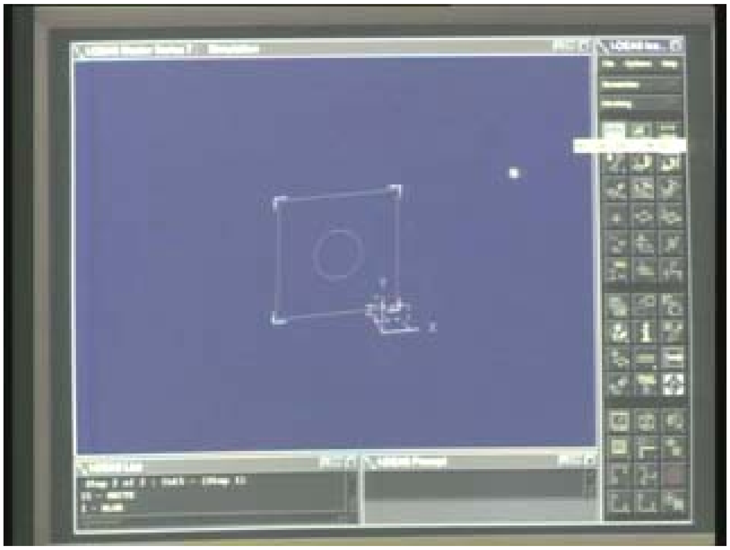
What he does now is to say that, you see that the color has changed. What he has done now is to say that this plane, which he has made by drawing the poly line, now becomes the working plane. In other words what he tells the computer is that whatever I am going to draw, please draw it in the plane in which I have just now drawn. That is what When he says that the color changes and that is where he is now going to draw the circle. Let us see how he draws the circle. Now, he has drawn the circle and what he does next is to extrude out or cut out from that sheet or this plane, whatever he has drawn now.

(Refer Slide Time: 16:35)



Yes; the cut out is now complete and now we get our original figure or the figure which is of interest to us which is nothing but a square with a circle at the centre. You now see that very clearly the plane is defined. These two lines define the plane, these two lines there define the plane and that the circle is eliminated.

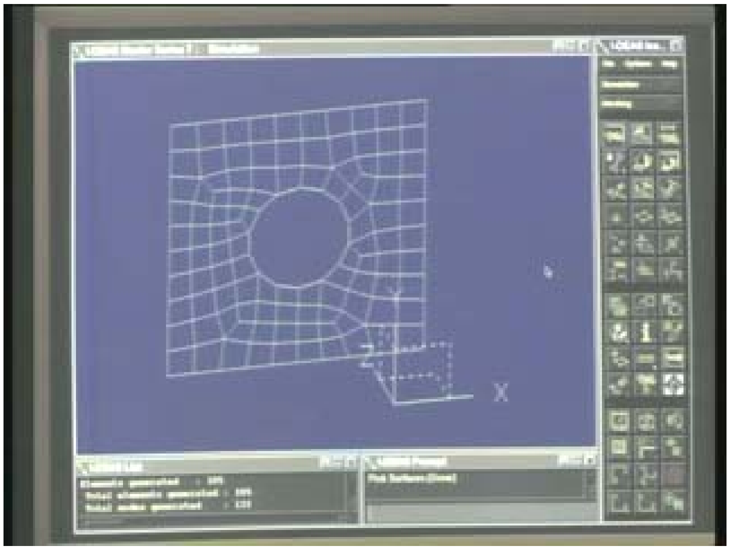
(Refer Slide Time: 17:03)



Having drawn this, what is our next step? We have to now define the mesh. Now this is drawn in one section of the software. There is a modular or what the ideas call it as master modular. They are called with different names and so on. So, by means of a modular he has drawn the geometry. Is it clear? What he does next? He has to take this geometry inside what is called as a preprocessor. The preprocessor helps us to put down or to define the elements. He is now going into the preprocessor. Let us see what he does. Yeah, we will go now to the preprocessor, which for example ideas calls it as a meshing package.

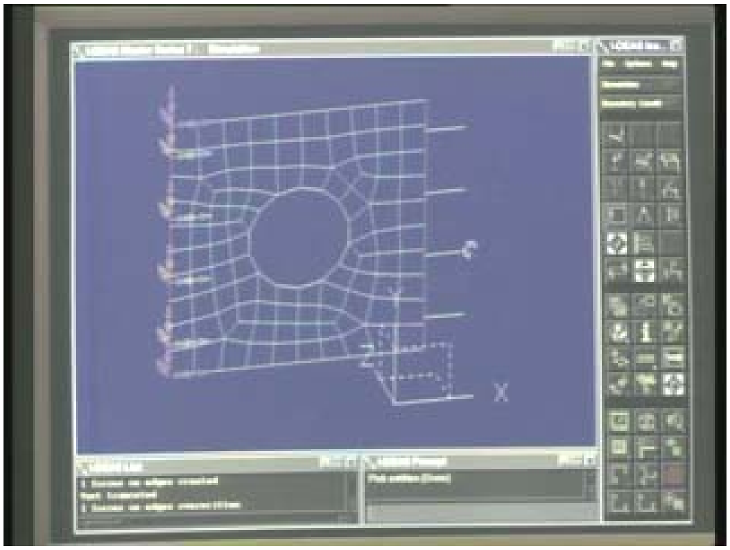
Now what he does is to define the mesh. It is asking for the type of mesh. He is now saying that the mesh is a shell mesh and gives a name to it. In this particular package shell element and plane stress elements are the same because shell is actually a super set. We consider it as a super set of plane stress and hence he is defining a shell element or in other words, I think the plane stress is itself is there. So, he is defining a plane stress type of element. You can also use a shell, because it gives better results actually for a cut out in the centre, because in the cut out in the center there can be out of plane displacements as well; mechanics wise there can be out of plane displacements. You can either use plane stress or better still you can use a shell element. Now, he has generated the elements.

(Refer Slide Time: 19:20)



He has defined the type of element. He has defined the geometry before that and he has now defined the elements. He has now got the elements. What is the next step he is going to do? He is now going to define boundary conditions. Let us see what he does for that.

(Refer Slide Time: 19:44)



In order to do that, he goes to another module where boundary conditions have to be defined. In that module he defines the boundary condition by pointing out the line where he wants to fix. That is that left hand side is that line where he is going to fix or in other words what he says to the computer is fix all the nodes which are lying on the line which is just now shown. You can see that he has fixed both X and Y displacement on that particular line. Now having done that, his next step is to define the load. Is that clear? Again he does the same thing, he does the same thing. He goes and says that he wants to define the load and then he goes and identifies a line where he wants this load to be defined and that is what he did by pointing out the right hand corner side of the mesh or the figure and then say what the amount of load is that has to be there?

In this particular package, it is possible for you to give the total load that acts. Then that load will be distributed through out. So, it is very simple to give a load and this load will be now transferred as the equivalent loads by consistent formulation as loads that are acting at the nodes. He now changes the direction so that the direction is consistent and now you can change the color of these loads, the displacement what you have given, boundary conditions you have given the screen, all these things you can do so that you can see it better and maybe, you can change the colors so that the white lines that you see, they are the loads that have been given.

Yeah, I think it is better that we change that color so that it becomes white. Yeah, he is now identifying that and he is now changing the color. It is okay because let us get on with it, but anyway nevertheless, you can see it. You can see it when he brings it near he can see you can see the white. Yeah, that is okay. Now having done these two, he has to now define the problem. He has not yet defined the problem completely or in other words he has to give a set of boundary condition and loads. It is very important to understand that in any problem you can define more than one set of boundary condition and more than one set of loads. In many practical problems it may be possible that you have more than one set of boundary conditions and more than one set of loads. For different conditions you can define like that.

For example, the same sheet, you would like to analyze it for bi axial tension. What is bi axial tension? That means you apply loads on either side in X as well as Y direction, but you would like to run it separately. So, the boundary conditions are different, loading is different. In most practical or most commercial packages, it is possible to define at one time a number of boundary conditions and loads and then when you run it, you just say that look I want you to run this problem where the boundary condition that you are going to use is a set called say 1, where I am only fixing the X and Y direction on the left hand side and take this load. You know, you can tell for the first analysis, I have two sets of boundary conditions, two sets of loads.

Take the first set boundary condition, first set load and run the problem. This is what you are trying to say. So, you create a set and that is what he is now doing. Now he says that take the restrain set, whatever I have given, even though you have created only one it does not matter, you have to say that I have created this 1, take that and then go to the load set that is at the bottom what you now see; it is highlighted. That is the load set. There is only one load set and one what is called as the boundary condition that you have given. Andrew is now telling the computer that, take these two for your analysis. Is that clear? So, it is possible to have more than one set. For example, you can have a boundary condition at that place at the bottom place fix it as well. You have not done it; it does not matter.

Now having done this, what is our next step? Our next step is to define material. It is possible that when you start itself you can have as a default, say a material, which is say for example steel. It is possible also to define a material which is different from steel. In this particular case, in this package, since we work quite a lot with steel there is a default material to be steel. So, as soon as you draw a part the package assumes that you are going to use steel. It is also possible to put aluminum. There is a material data that is there. We will see that in a minute, the kind of material data that you can use. For example here now it is isotropic steel but you can add if you want other material data as well by giving the properties, like for example this menu defines all the properties. There are more than 20-30 properties.

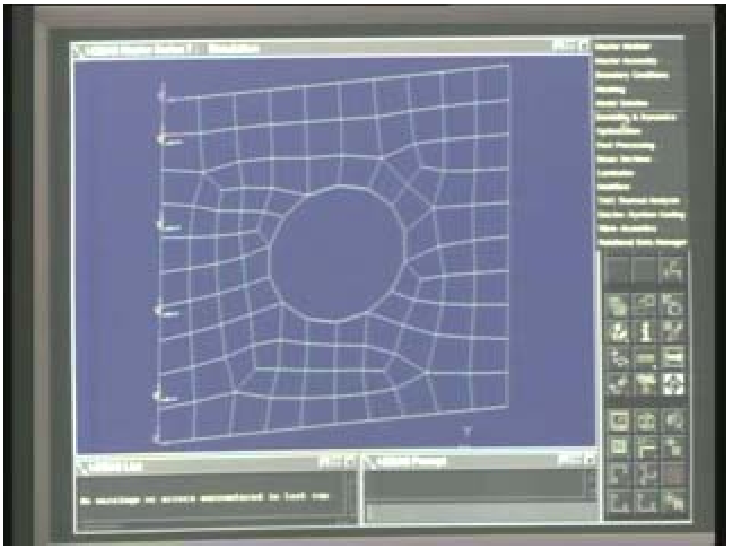
(Refer Slide Time: 26:25)

Every property you can do, because you can do thermal analysis and hence thermal properties based data are also available there. You can define these material properties, store it before hand, whether it is aluminum or steel or whatever it is and then you can call it when you do a problem and say that look the material that I am going to use is steel or aluminum or whatever exotic material that you would like to do. Now for the time being let us stick to steel and then let us now do the analysis. So the next step now so having What are the things we have defined? We have defined the geometry; we have defined now, the elements. We will go closer into it later, but we have defined the elements, we have defined the boundary conditions, we have defined the load and we have defined the material. Everything has been defined.

Let us now submit, as they call it, submit this particular job to do analysis. He has defined what all sets that he wants now. Before that he also has to tell two things. One is what is the type of result he wants? There is no point in calculating everything. May be you are interested in strain and stress, may be some times you are interested in stress strain, strain energy, whatever it is, because in order to check the quality of result you may look at strain energy and so on. It is possible at this point of time to tell the software that, look I want these results; I want strain, I want stress and so on. That also he has done. He has now done that, telling the computer what are the results he wants. Having done all this, he is now going to submit the job for running. Now, he has submitted the job. The modules change; so, you see that there is a change of screen, modules change, it now goes to solution module. Please note this. This is very important to realize that first we started with the geometric module, we did this geometry. Then we went to what is called as pre processor module, we created a lot of things there meshing and all that. Then we go into what is called as the solution module. Now, he has gone into the solution module, where he has now submitted the job for solution. It will take a couple of minutes.

After completing the solution, we will go into a post processor module. Now see, it gives out what are the steps that it is doing? All those steps like for example it says that I am solving this algebraic equation, set of simultaneous equations, after all what is that we have done? A differential equation, partial differential equation we have converted into an algebraic or a simultaneous equation, Kd is equal to F. It forms stiffness, it assembles; all this review you will not be able to read it very clearly but that is the message which it says.

(Refer Slide Time: 29:35)



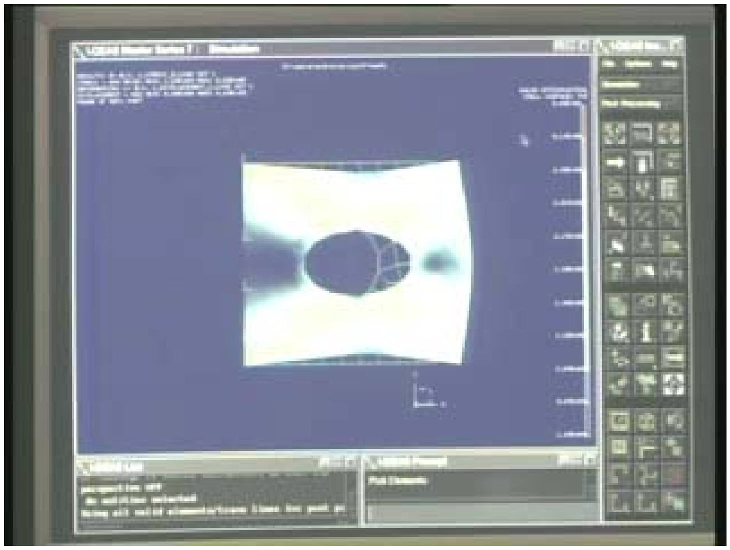
Yes, now it has done all the things. The results are stored. Let us look at the results now. We are now in the post processor module, so, please note the following things. One is that we do not This is the confusion always which happens; I have written so much about theory. We do not, when you use the package, calculate stiffnesses and so on; the package does all these things. But what we require is a knowledge which I am going to illustrate by a next simple problem. Whatever we are learning now is required in order to run any practical problem. You should know the type of element you are using. It is a very thin sheet, so, I used plane stress.

If I want more accurate results, actually I should use shell, basically because at this position there can be out of plane displacements as well. At the location near this hole, there can be out of plane displacements in which case plane stress will not show. Nevertheless, that is the first problem. It is okay if we do; it has a plane stress because we are only looking at the steps that we are going to go through, in order to do this problem. Now, let us look at the results.

Let us look at say the stress as well as displacement. But, you see there that, may be you are not able to read it, nevertheless what is called as the Von Mises stress. We have studied quite a bit about Mises stress and Von Mises stress is what is written there. In fact you can get lot of other things. If you really see the screen, it is written that there is a maximum principle stress, middle principle stress, the minimum principle stress, the maximum shear, Mises $sigma_{11}$ $sigma_{22}$ $sigma_{33}$ $sigma_{12}$ $sigma_{13}$ and $sigma_{23}$. These are the things that you can get completely from the post processor. Apart from that, we can also look at the displacements or deformation; that also has been done.

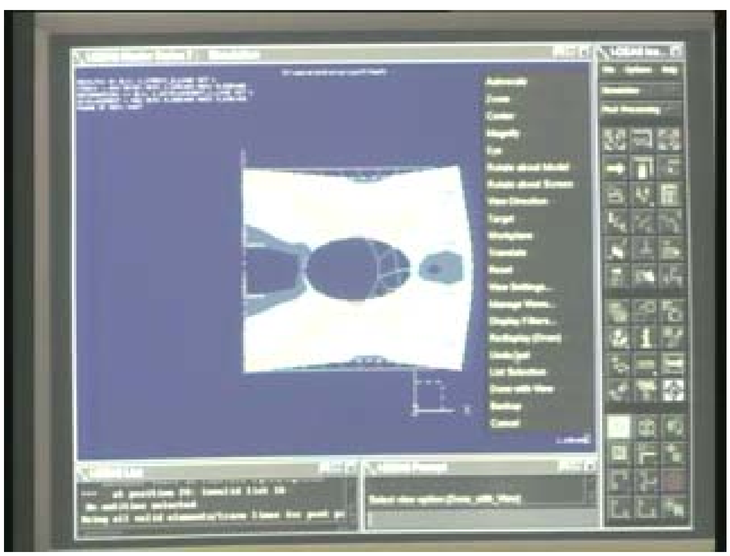
Here also you can see that X Y and Z displacements can be seen. For example let us look at X displacement or we can look at the displaced magnitude of it. Let us look at that and see how the results are. Now it is processing it.

(Refer Slide Time: 32:19)



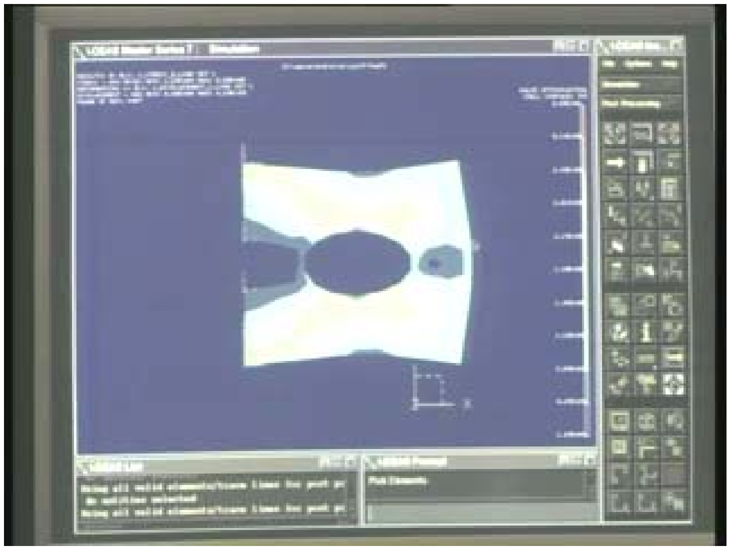
Yes; this is the displacement. You see the maximum and minimum displacement. I think it just gives the magnitude, but the y direction displacement and we see that the maximum stresses are there at these positions. Is that clear? Let us go back to our results. There are two things that it asked, a step shaded and smooth shaded. Let us have a look at that. You can see that he is now changing the result to be displayed as step shaded and let us look at the result there.

(Refer Slide Time: 33:17)



You can see a big difference in the way actually the material or the stresses are displayed.

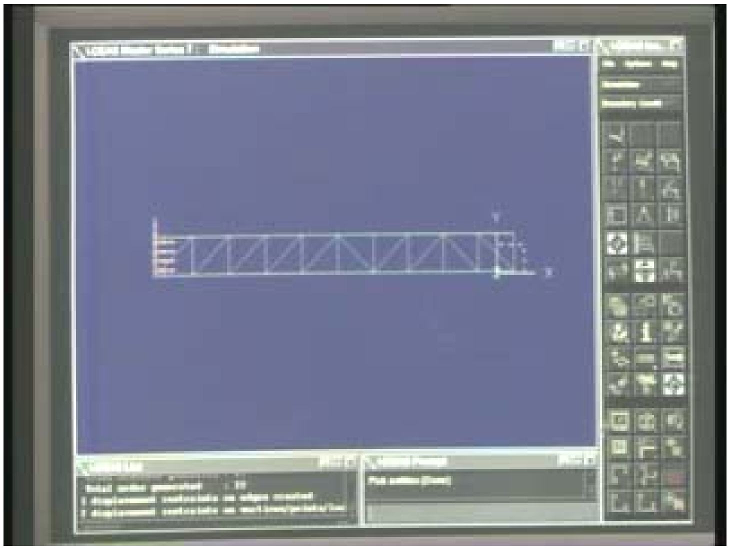
(Refer Slide Time: 33:24)



You can see that there is a gradation, which means that the results are not definitely smooth. On the other hand, you can see that the other result that you saw before is smooth shaded. Again what does it mean? It again means that, since there is a stress jump, how the interpolation is taken into account at the nodes may be different. You can average out or better still that is what is called you can have a special scheme in order to get the stress at the nodes. We will see this later in the course but I just want you to observe that there are different ways in which stresses can be looked at or can be what we call as represented, because of this kind of variations in stresses as you move from one element to another element and so on. We already noticed that there is a stress jump. Let us take another problem and let us take straight away a beam problem and look at the problem as to what are the results that you get.

Now you understand, I hope, the steps that are involved in order to do a problem. He is now going to close down this and then do a beam problem. Let us see what are the steps that are involved?

(Refer Slide Time: 35:01)

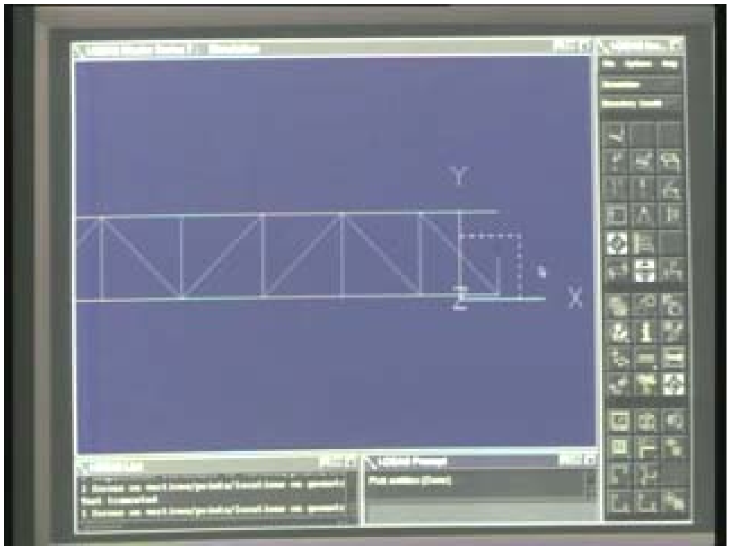


Again we will do it as a plane stress, but this time we are going to use triangular element. He now will vary the number of elements that he is going to use across the thickness and let us see how the results would vary. Yeah, he is making it into one plane, because they are curves. When he will do it as poly lines they were independent curves. Now, here he has made it into one plane by saying that the plane is bounded by these curves. Just a second, we will just change the dimension; the length is 100 and the depth is 10. The length is 100 and the depth is 10. The usual practice of most people who work in CAD is to just draw a figure, then go and change the dimension. That is what most of the people do and that is the easiest way to do as well; just draw the figure and then keep changing the dimensions and that is what here also we have done, so, 100 and 10.

Now, he is making it into a plane, he is going into the meshing part of it. We will do a plane stress element and a triangle. Now, it asks for what is called as element length. So, we will give a big element length so that we will get a few elements. Fine, so, let us now see the mesh. You can at this point of time, if you want, change the mesh also. We will just stick to this. We will see the effect of meshing on the result. That is what we are going to see. Especially we will watch out for displacement and maximum stress; this is what we will watch out.

What is the other next step? We have already seen all the steps. He is going to define the boundary conditions. He is going to fix x and y at that place and then he is going to give a vertical load of say 100 in the negative direction.

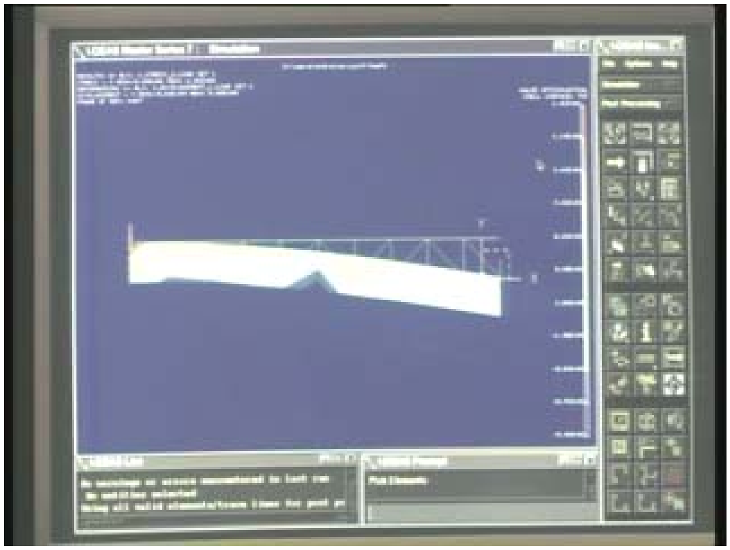
(Refer Slide Time: 38:52)



You see that; that line there that load is given at that node and that is the load that he has given. He is now doing all the things, which he did for the first problem. He is defining the load set and the stress strain - ? set to be taken by the computer. Those two he has done and he has gone to the solution and what are the results he wants? All those things he has given and now the solution is going on. Yes, the solution is over. Now, he has to move to the other module, say, post processor module and let us look at the results.

Let us look at the stress and say we will let us look at sigma$_{11}$ that is a bending stress and the displacements, displacement in the y direction.

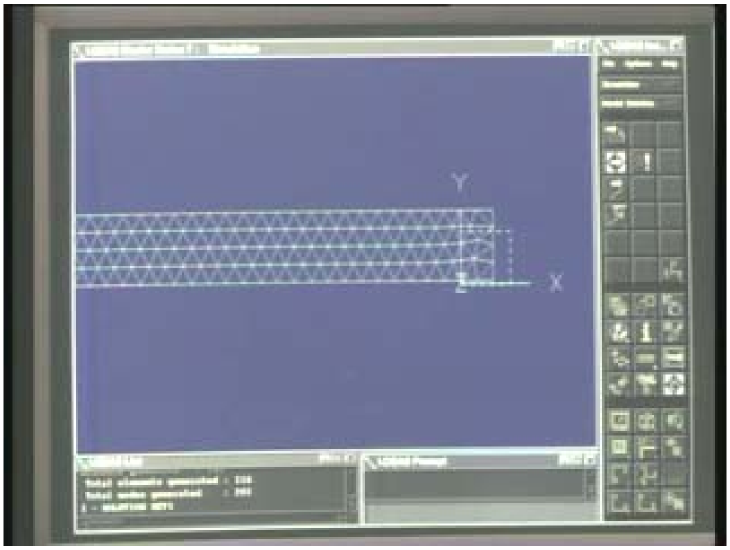
(Refer Slide Time: 40:41)



Just note down the values which I am going to give; may be I will also note down the displacement. The maximum displacement at the end is 4.66 into 10 to the power of minus 4; 4.66 into 10 to the power of minus 4 and the maximum stress, yeah, let us not worry about the units now. Put a 10 to the power of 2 milli Newton per mm square. 1.41 into 10 to the power of 2 is the maximum stress at the end where reactions are taken.

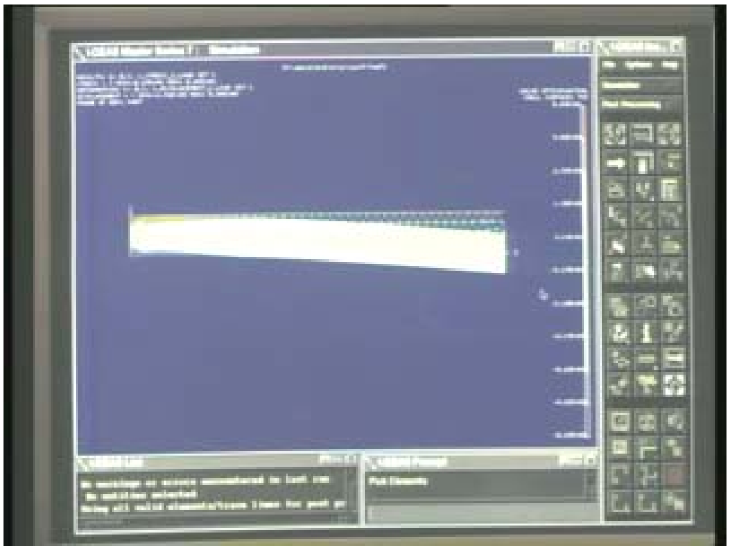
Let us get to that same problem, refine the mesh. Let us make the mesh say four times better than this. Let us see whether there is an effect. Do you think there will be an effect? Yes, let us see if there is an effect. We will change the element length, may be make it 2, yeah okay, 2.5. We will define the load, define the load at that point and see; the same load 100.

(Refer Slide Time: 42:10)



What he has essentially done is to remove the load, may be he has to remove the results also for this in order that he can run this problem. You cannot run a problem with the results on, because it does not know; computer software does not know whether to delete the results or rewrite it at the same place. Now, he has redefined the nodes and then he is now running it. Let us look at the same results for this problem, stress and the displacement. That is stress, x stress and the displacement in the y direction. Let us see what happens to the result.

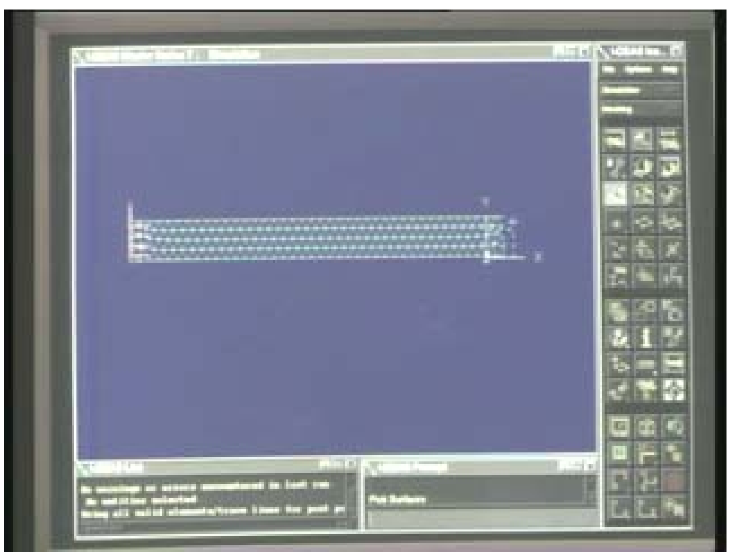
(Refer Slide Time: 43:00)



You can see that the displacement is drastically different, both stress as well as displacement are drastically different. The stress now happens to be 5.06 into 10 to the power of 2 when compared to 1.41 into 10 to the power of 2 and the displacement is also drastically affected, it becomes 1.72 into 10 to the power of minus 3 and this very clearly shows that you cannot just take an element, just take a size, just run it and so on; you may be totally wrong unless you understand the problem. What is that difference? Please note that just now we had a small discussion. We said that the triangular element which we have used is a constant strain triangle. But in this bending problem, the stresses vary right across this and so that element is not a very nice element to do this problem. It is very important that it is not just you understand the menus in the steps. It is also important what goes behind this whole thing.

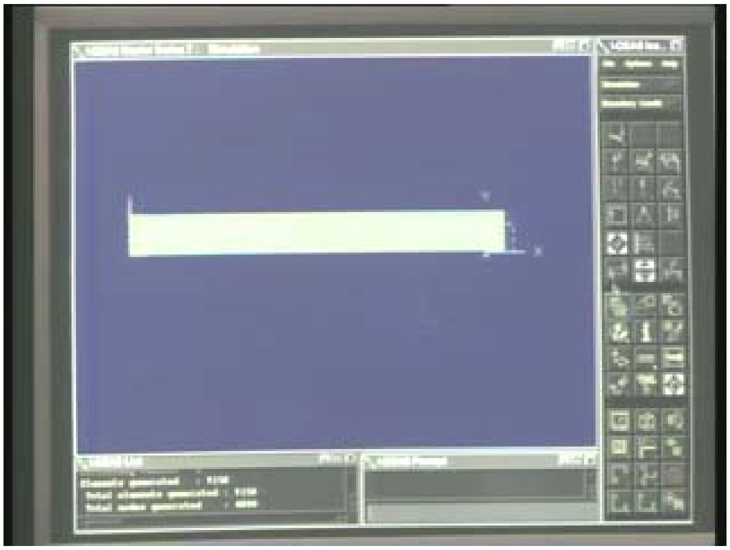
Let us take a finer mesh, much finer than what we have done and may be it will take a couple of minutes, but we will run the problem again and see what the result will be.

(Refer Slide Time: 44:27)



He is now deleting the solution so that the software can freshly write, because he is not coming out of the particular file. You cannot have more than one result, so, he is just deleting it and now he is again redefining it. For example, he is going to give an element size of 0.5; much finer. There are 436 elements, sorry 436 nodes and nearly 9150 elements. Let us look at the, yes, fantastic.

(Refer Slide Time: 45:01)

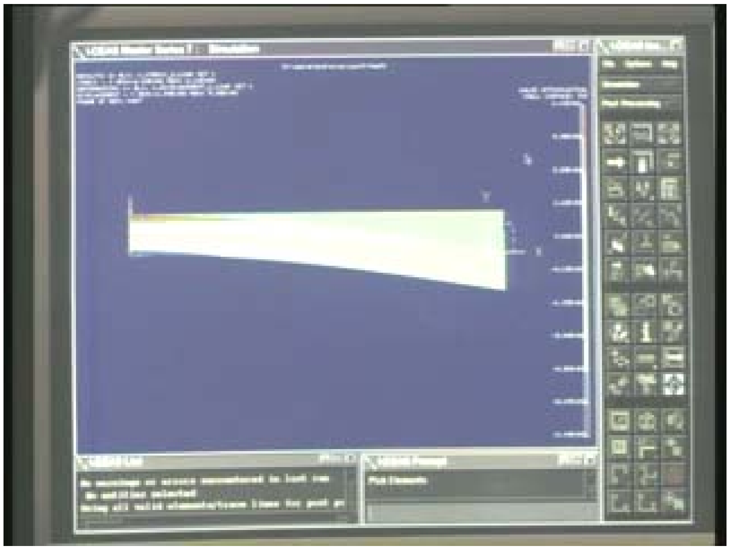


I hope we will be able to run it within our allotted time. Let us see and let us look at the result. He will define again all the things, the load and other things. The beauty of it here is that you need not every time go and select the nodes which lie on the left hand side, which lie at this point. It is not necessary that you go on to define all this; just say that all the nodes in that line, the software would automatically look at the nodes which lie on that particular line and go and arrest it. He has done both and let us see what the result is now. It is running and it is going to take slightly more time and let us see what the results are.

(Refer Slide Time: 45:54)

Yes, it is showing the time taken; the time that could be taken for doing this problem. It gives you a sort of rough indicator that this is the CPU time that it is going to take. Yes, I think the solution is complete and let us now look at the displacement and the stress.

(Refer Slide Time: 46:32)



Yes; now, let us look at the stress. The stress is 6.16; 6.16 into 10 to the power of 2. Last time how much time did we have? 5 point; so, you see that it is converging. What did we do? We made it one-fifth, but still the stresses have not increased that much. Last time we made it one-fourth, there was a drastic change. Now we changed it, the stresses have changed, but not to the same extent as it happened for the first time. You can see the displacements also. Displacements happened to be 1.94 into 10 to the power of minus 3 and last time it was 1.72. So, you can see that a convergence come about as we make the mesh finer.

We can do this, this kind of convergence study, but even to do that you should understand how the element behaves. The major warning, one of the reasons why I wanted you to show this, I wanted you to see this, is that just like that you cannot choose an element, put some element size, just put the loading, get the result and so on; you may be totally wrong. The results what you get may be totally off; 200 to 300% off and hence it is important that we understand what we are doing and do the problems. We will stop here. We will stop here for this class and we will continue with some more theory and may be in a later class, we will have more clarity in order to look at these things again. We will do a series of problems later in the course.